Numerical Study and Simulation in COMSOL Multiphysics of the Dilution Process during Dust’s sampling in Dry Machining.

Wenga Ntcheping1*, Kamguem1, Kouam1 and Songmene1
1University of Quebec/École de technologie Supérieure/Mechanical engineering/LIPPS
*Corresponding author: 1100, Notre-Dame Ouest Montréal (Québec) H3C 1K3, bernard.wenga@yahoo.ca

Abstract: The importance of dilution’s issue during the dust’s measurement has risen in recent years with the interests of researchers and engineers working in environmental and occupational safety sector. In fact, the study of machining process recommended adequate devices with best sampling. Furthermore, the excellent measurement of fine and ultrafine particle passed through the dilution of high concentration. The aim of this paper is to propose a numerical study and graphic simulation for the best choice of dilution’s issues during dry machining. The constitutive Navier-Stokes equations and CFD k-ε model was used in COMSOL Multiphysics to obtain the graphic simulation of dilution and choice the alternative which was the inlet and outlet in the same jet direction. The experimental setup confirmed the mean of dilution ratio with 95% confidence.

Keywords: Mixing, Simulation, COMSOL Multiphysics, Dilution factor, Dry machining.

1. Introduction

Metallic’s particle was considered as the principal responsible of health effects [1, 2] caused by airborne particles in workplace. The quantitative evaluation of fine and ultrafine particle emitted during dry machining was seriously interested by researchers. In 2002, Zimmer &al. have studied the influence of operating parameters on the number-weighted aerosol size distribution in arc welding [3], and grinding [4]. Songmene &al. [5, 6] quantified the PM$_{2.5}$ concentration during drilling for various materials (6061-T6, A356, AZ91E, 70-30 Brass) and different cutting conditions (cutting speed, feed, metal removal rate...). Khettabi &al [7-10] in turning, Zaghibani &al [11] and Kouam &al [12] in milling permitted to evaluate the high level of particle concentration in the experiment. Their works proofed the badly influence of high concentration in the mixing and the maintenance of air quality device (Laser photometer_DUSTTRAK, Aero-dynamic Particle Size Spectrometer_APS, Micro-Orifice-Uniform Deposit Impactor_MOUUDI, Electrical Low Pressure impactor_ELPI, Scanning Mobility Particle Sizer_SMPS, Light Detection And Ranging_LIDAR, Condensation Particle Counter_CPC, Differential Mobility Particle Sizer_DMPS) available in the laboratory.

To reduce these effects, the dilution issue was proposed in this study. A simple diluter adapted to the sampling during machining was designed in CATIA V5 and linked in COMSOL Multiphysics to simulate the dilution process. The present paper was organized in six sections: Governing equations; Methods; Theory and numerical model; Experimental results; Discussions; and Conclusion.

2. Governing equations

The Navier-Stokes constitutive equation was used to modeling the fluid transport and the mixing rate. The constitutive equation was composed of continuity equation (eq.1), momentum equation (eq.2), and energy conservation equation (eq.3) [24]:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \rho u_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \begin{array}{c} \rho \frac{\partial u_i}{\partial x_j} \end{array} \right) \tag{2}
\]

\[
\frac{\partial \rho H}{\partial t} + \frac{\partial \left( \rho (H + P) \right)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \begin{array}{c} \rho \frac{\partial u_i}{\partial x_i} \end{array} \right) + \rho g_i + \frac{\partial}{\partial x_i} \left( \begin{array}{c} \frac{\partial \tau_{ij}}{\partial x_k} \end{array} \right) + \frac{\partial}{\partial x_i} \left( \begin{array}{c} \frac{\partial q_i}{\partial x_k} \end{array} \right) \tag{3}
\]

Where: \( H \): total enthalpy per mass unit, \( S \): source of heat, \( \tau_{ij} \): viscous stress tensor, \( g_i \): gravity acceleration, \( q_i \): diffusion heat flow, \( \rho \): volumetric mass of the air, \( P \): pressure in the mixture, \( u_i \): displacement.
3. Methods

The dilution phenomenon was defined as fluid mechanic problem which could be solved using combined analytical and numerical methods. Generally, it’s usually difficult to solve the Navier-Stokes equation modelling the mixing fluid with turbulence. The Computer Fluid Dynamic (CFD) was proposed to solve numerically the problem and simulate it graphically. The 2D or 3D design of diluter was made in CATIA V5 and live-linked in COMSOL Multiphysics [25]. The experimental dilution in COMSOL Multiphysics was used with the following assumptions:

- Inlet boundary conditions were set with the physical geometry of diluter;
- Aerodynamic diameter of particle is less than 2.5μm;
- Properties of the fluid (air) were determined at 25°C with atmospheric pressure;
- Inlet velocity of pollute and clean air have been imposed by experimental conditions;
- Fluid was considered incompressible;

4. Theory and numerical model

With the numerical method, the constitutive equation was solved and solution convergence was given with k-ε model [26] represented by the two following equations (eq.4, eq.5): k the turbulent kinetic energy and ε the dissipation rate of turbulent energy.

\[
\frac{\partial k}{\partial t} + \frac{\partial k u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right] + G_k + G_p - \rho \varepsilon Y_m + S_k \tag{4}
\]

\[
\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \frac{\mu + \mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right] + C_\varepsilon \frac{\varepsilon}{k} (G_k + C_{\varepsilon} \varepsilon) - C_\varepsilon G_k - C_\mu \varepsilon^2 + \frac{\varepsilon}{k} S_\varepsilon \tag{5}
\]

- \(G_k\): kinetics energy due to the velocity gradient;
- \(G_p\): \(k\) energy due to detachment and volume forces; \(Y_m\): contribution of the expansion fluctuation in turbulence for \(\varepsilon\); \(C_1=1.44\), \(C_2=1.92\), \(C_3=0\), \(C_{\varepsilon}=0.09\); \(S_k\), \(S_\varepsilon\): source terms; \(\sigma_k\), \(\sigma_\varepsilon\): turbulent Prandtl number for \(\varepsilon\) and \(k\), \(\sigma_k=1\), \(\sigma_\varepsilon=1.3\).

Triangular meshing [27] was applied with 18846 elements and providing 89368 degrees of freedom. The momentum equation was implemented by a stationary nonlinear solver [25] using generalized minimal residual method (GMRES) was chosen to minimize memory consumption of computer.

Figure 1. Meshing of the diluter in COMSOL

The dilution process was implemented in COMSOL Multiphysics to firstly observe the evolution of the mixing in two alternatives and choose the practical alternative adapted for experiment in workplace.

Secondly, the reason justified the numerical was the location of adequate test in the choice’s alternative which was favorable for best dilution.

5. Experimental results

COMSOL Multiphysics permitted to obtain picture of dilution’s simulation which could visually analyse by experimenter. The different color of these pictures presented visually the level of dilution. The vortex and the turbulence phenomena were directly observed in the computer screen.

Figure 2. Picture of velocity magnitude simulated in COMSOL during dilution
The singular evolution of the test done in COMSOL Multiphysics, was observed in table1. This table present the test in two alternatives: Alternative «A» presented the situation where the two inlet jets and the outlet flow were in the same direction; Alternative «B» has shown the situation where the two inlet jets were in the opposite direction.

Table 1: Pictures of dilution’s process in COMSOL

<table>
<thead>
<tr>
<th>Test</th>
<th>Alternative A</th>
<th>Alternative B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The different tests have shown that alternative «A» was more excellent in dilution process than alternative «B». The number of vortex directly observed has confirmed this situation of best turbulence in the mixing rate.

6. Discussions

The number of vortex was the principal reason of best dilution because of relaxation’s effect during the entrance of two jets (inlet clean air and inlet polluted air) in the cylindrical chamber of diluter. The relaxation and turbulence were due to the wide variation between the small diameter of inlet tube (4 mm) and great diameter of the chamber (40mm). This geometry increased the turbulence and easily mixing air and particle. In addition, the recirculation of flow mixture remains low even when the two inlet speeds are equal. The two-phase of mixture] seems to follow in the chamber so as to obtain the desired dilution for a possible extent. However, concentration level was represented by the type of color presented (light blue or dark blue). It is quite clear that direct observation of the simulation’s picture could help the experimenter in the dilution process. A comparative study was made on both alternatives where alternative «A» presented the best result in the dilution process. In this case, the alternative «A» was chosen by experimenter to validate in the dilution process during dry machining in the workplace. The dilution process was tested statistically in the workplace where the mean value of dilution factor was obtained with 95% of confidence. This final result was applied with good appreciation during sampling of metallic particle in dry machining.

7. Conclusion

This research’s work concerned the numerical study and the simulation of dilution process during dry machining. The CFD k-ε model was used in COMSOL Multiphysics to obtain the graphic simulation of dilution which permitted to choose the best alternative where inlet and outlet test done during dry machining confirmed the simulated mean of dilution factor with 95% confidence. In future work, the dilution models could be discussed and implemented in COMSOL Multiphysics to make a comparison.

8. References


8. Appendix

Table 2: Numerical test of dilution in COMSOL

<table>
<thead>
<tr>
<th>Test</th>
<th>Polluted air</th>
<th>Clean air</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Speed(m/s)</td>
<td>Flow (cc/min)</td>
</tr>
<tr>
<td>1</td>
<td>0.0</td>
<td>00000000.0</td>
</tr>
<tr>
<td>2</td>
<td>0.5</td>
<td>3769911.2</td>
</tr>
<tr>
<td>3</td>
<td>1.0</td>
<td>7539822.4</td>
</tr>
<tr>
<td>4</td>
<td>1.5</td>
<td>11309733.6</td>
</tr>
<tr>
<td>5</td>
<td>2.0</td>
<td>15079644.8</td>
</tr>
<tr>
<td>6</td>
<td>4.5</td>
<td>33929200.8</td>
</tr>
<tr>
<td>7</td>
<td>3.0</td>
<td>22619467.2</td>
</tr>
<tr>
<td>8</td>
<td>3.5</td>
<td>26389378.4</td>
</tr>
<tr>
<td>9</td>
<td>4.0</td>
<td>30159289.6</td>
</tr>
<tr>
<td>10</td>
<td>4.5</td>
<td>33929200.8</td>
</tr>
</tbody>
</table>

7. Acknowledgements

The authors would like to acknowledge the financial support of Products, Processes and Systems Engineering Laboratory (LIPPS): located at «École de Technologie Supérieure» / University of Quebec-Montreal Canada.